AN INVESTIGATION INTO NAVIER-STOKES SOLUTIONS OF THE FLOW AROUND WING-BODY GEOMETRIES

By A.S. Jonker, P.W. Jordaan, J. Bosman, R.S. Neethling

Presented at the XXVI OSTIV Congress, Bayreuth, Germany

ABSTRACT

In this paper the commercial CFD packages Star-CD and Flo++ was used to investigate the suitability of the Navier-Stokes method as a general aerodynamic design code. The packages were used to model a simple wing-wall geometry and the wing-fuselage junction of a hypothetical glider. The normal problems assicated with Navier-Stokes codes, i.e. grid generation, solution time and accuracy were investigated. It was found that the problems of grid generation and solution time have greatly been solved by a dedicated CAD interface and the availability of faster computers. The accuracy compared to experimental results was, however, lower than expected. This is due to the fact that only a simple k-∈ turbulence model was available for the commercial codes used. The Navier-Stokes method will be suitable as a general design code for glider designers as soon as better turbulence models become available.

INTRODUCTION

The performance reached by modern sailplanes is very high. The maximum glide ratios of 15m class gliders are around 45, while the open class gliders have reached 60. This is the result of careful aerodynamic design, using modern computational techniques as well as wind tunnel testing. Almost all components have been optimized to a point where very little improvement is possible. The wing-fuselage junction is, however, a region where the possibility for improvement does exist[1]. This is due to the fact that the flow in this region is very complex and not yet theoretically predictable with high accuracy[2].

The most commonly used computational techniques in aerodynamic design are the panel methods. These methods are based on the potential flow equation that can be derived from the Navier-Stokes equation with the help of several simplifying assumptions[3]. Panel methods, as a result of these simplications, are very good at predicting the flow where viscous and compressible effects can be ignored. However, these methods fail to give accurate answers in the case where the flow is separated, as in the case with a wing-body junction. An advantage of the panel methods is that it requires calculation only over the surface of the body instead of the full three dimensional flow field, as with other computational fluid dynamics methods. The calculation is therefore only two dimensional which requires must less computational effort.

The solving of the Navier-Stokes equations for three-dimensional flow requires much more computational effort because the flow must be solved for the complete three-dimensional flow field around the body. The time required to generate the grid and the required computational time previously prevented this method from becoming a practical design tool. With the availability of faster computers the computational time required has dropped dramatically. The development of a CAD interface designed to simplify the design of glider geometries has also simplified the generation of the computational grid. This leads to the point where Navier-Stokes methods might be considered a practical design tool for the design of high performance sailplanes.

The purpose of this papr is to investigate the use of a commercial Navier-Stokes code as a practical design tool. The code will be used to analyze the flow around a wing-body junction of a 15m sailplane. The effort required to generate the grid, the computational time as well as the accuracy of the results will be investigated.

INVESTIGATION METHOD

In this study the problems normally associated with Navier-Stokes codes solutions will be investigated. This will be done by the process of modeling complex flow fields where viscous effects and separation are present such as wing-fuselage junctions. As a first step the flow field around a wing-wall geometry will be investigated. This problem is easier to model and some experimental data is available. A model for the flow field around a hypothetical 15m class glider will then be developed. This model is much more complex as the wing root fillets will be modeled. This problem will give some insight into the problem of grid generation and solution times. The commercial Navier-Stokes codes, Star-CD and Flo++ will be used. The codes will be used as provided by the developers without any user coding or special models, which will give a better indication of the suitability of commercial Navier-Stokes codes as a design and analysis code for glider designers.

THE NAVIER-STOKES SCHEME

In the investigation of a flow region where laminar as well as turbulent flows are expected, it is necessary to use a method with turbulent flow capabilities. The unsteady Navier-Stokes equations (conservation equations for mass, momentum and energy) are generally considered to govern turbulent flows. The direct numerical solution (DNS) of turbulence with this set of equations is however not possible because of the huge number of grid points and the small time steps that would be required to resolve the small space and time scales of turbulent motion. The computational effort increases with the cube of the Reynolds number.[4] For the practical solution of turbulent flows another set of equations are considered, namely the time averaged Navier-Stokes equations, also called the Reynolds equations of motion. In these equations, turbulent motion is described in terms of time averaged quantities rather than instantaneous. A turbulence model is thus always required to close the Reynolds equations if turbulent flow is modeled.

Current commercial codes use the k-∈ method. Although this method is not totally suitable for the modeling of external flow fields such as wing fuselage junctions, it is the only model currently avaialable for the commercial code used in this investigation.

There are three problems in using Navier-Stokes codes as a practical design code:

- The flow must be solved for the complete three-dimensional flow field around the body which implies a very involved grid which is very time consuming to generate.
- As a result of the large number of grid points in the three-dimensional flow field, the solution is expensive in terms of computer time.
- Turbulence models that are generally available, are not very well suited for complex flow fields as found at wing-body junctions.

These problems will be investigated in the course of this paper.

CASE 1: THE FLOW AROUND A WING-WALL GEOMETRY.

The first case to be modeled is a wing mounted to a wall as shown in Figure 1.

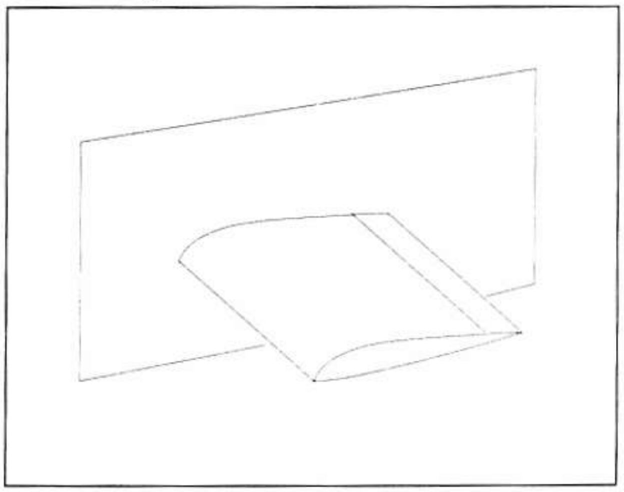

Figure 1: Wing-wall geometry.

This geometry was chosen because it is easy to model and it could be compared to wing-tunnel data. The airfoil section used, is the AS97-129/14 which is very similar to the DU89-134/14, which is used on the ASH26 and ASW27[5] sailplanes. XFOIL[6] pressure distribution for this airfoil is shown in Figures 2 and 3.

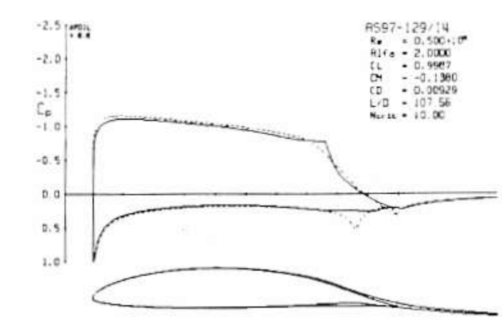

Figure 2: AS97 Pressure distribution for α = 2°

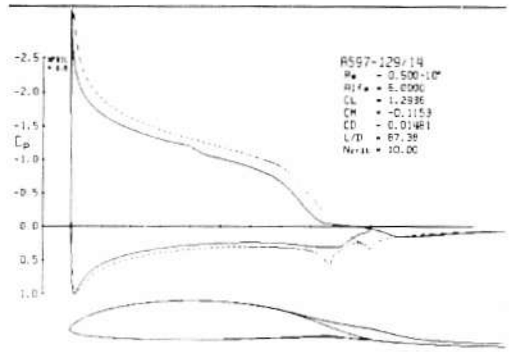

Figure 3: AS97 Pressure distribution for α = 6°

THE COMPUTATIONAL GRID

A structured computational grid of approximately 300,000 cells was used to model the problem. It is basically a rectangular block with the wing section cut out from it as shown in Figure 4. The grid is refined towards the wall to ensure high accuracy in the region where the expected wall effects will be present. Figure 4 also shows grid refinement around the airfoil surface to enable the boundary layer effects to be captured.

The grid is fully parameretized to allow changes in the angle of attack or grid refinement to be performed very quickly. Initial development time was around 3h while regenerations of the grid can be performed in a matter of seconds.

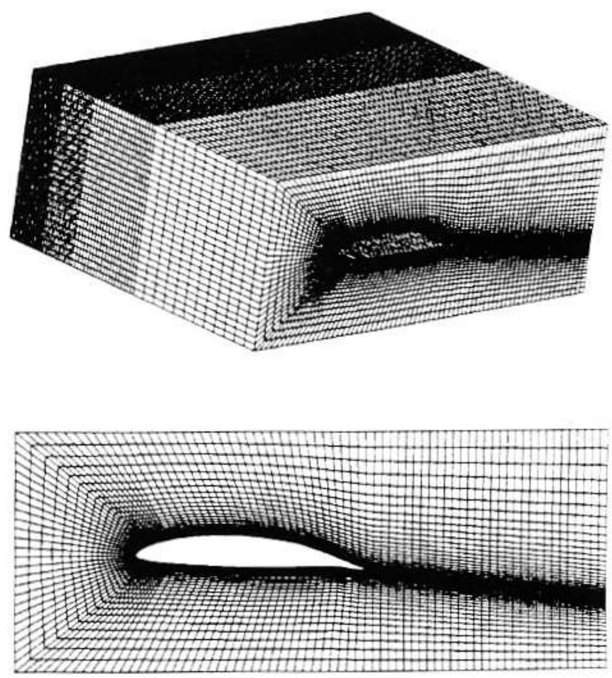

Figure 4: Computational grid for wing-wall problem

Pressure boundaries were used at the top and bottom to prevent wall effect, while a wall boundary was used to model the wing-wall junction. The opposite junction is modeled with a symmetry boundary. The airflow is constant across the inlet.

RESULTS

The computation was done on a 300Mhz Pentium 2 with 256 MB as well as a DecAlpha workstation. The solution times are shown in Table 1. The PC took approximately 4.5 h to complete the calculation while the workstation took only 2.5 h. These times are relative high, but can be considered acceptable in the light of the complexity of the problem and the time wind tunnel testing will take. The promise of a 1000MHZ PC towards the end of 1999 will decrease times even further and yield the method even more attractive.

Table 1: Processing times for Case 1

Processor	Number of cells	Time to convergence (s)
Pentium 2 300Mhz	318 000	16808
DecAlpha	318 000	9369

Navier-Stokes code normally gives the velocity strength and direction as well as the pressure at each cell as a result. The lift and drag, which is important to aerodynamicists, can be calculated from the pressure data. As this necessitates user written codes which require considerable effort, it was not done for this study. The velocity direction and strength were used to identify recirculation zones where the flow separated from the wing-wall surface and it was compared with data from a flow visualization performed in a wind-tunnel. The pressures on the top and bottom surface were compared with the data from the two dimensional airfoil design and analysis code XFOIL, developed by M. Drela[6].

Figure 5 shows the results of the flow visualization for an angle of attack of 2 degrees at Re of 50,000. The flow away from the wall is laminar up to about 70 percent chord where natural transtion to turbulent flow takes place, which agrees with the XFOIL results in Figure 2. The dark strip on the right hand side of the wing in Figure 5 is a trip strip. Next to the wall the flow separates ahead of the wing due to the adverse pressure gradient as a result of the presence of the wing. The vortex downstream of the separation wraps itself around the wing to form a horseshoe vortex. This is visible as the slightly lighter region next to the wing section. Further downstream the vortex separates from the wing to form a clearly visible fishtail. Figure 5 also shows the flow to separate from the wall. An interesting separation pattern is shown on the trailing edge of the wing next to the wall.

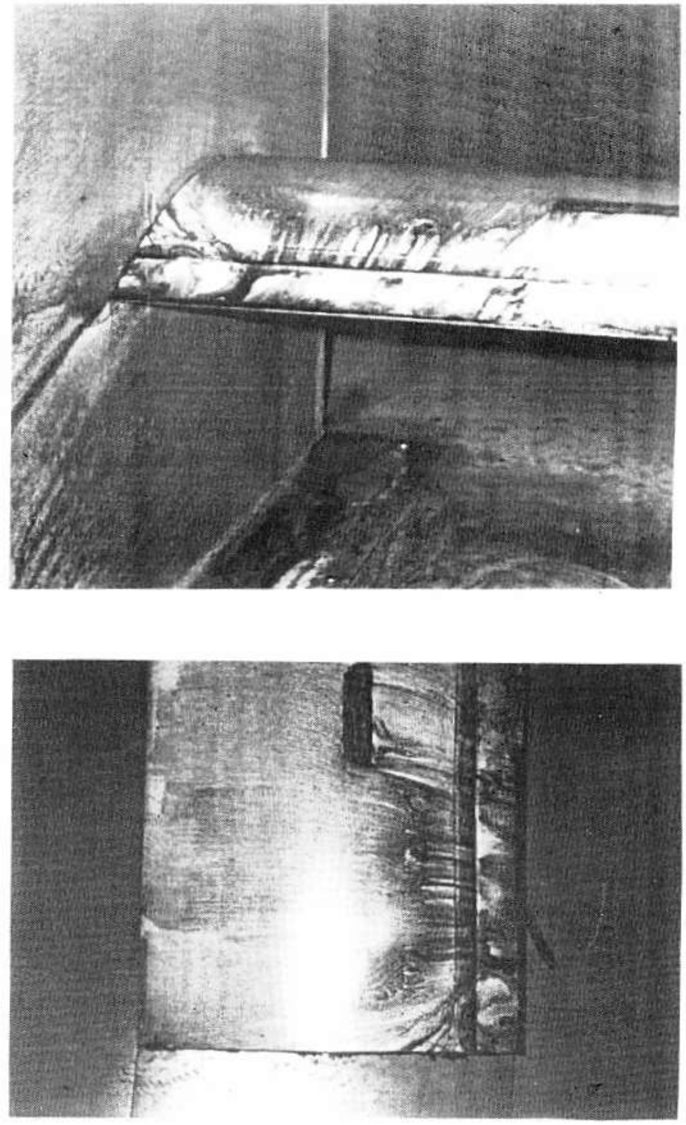

Figure 5: Wing-wall flow visualization.
$\alpha = 2^\circ$ $Re = 500\ 000$

Figure 6 shows the oil flow patterns for an angle of attack of 6°. The flow is completely turbulent with separation at approximately 80 percent chord. This is in agreement with the XFOIL which predicts flow separation at 80 percent chord with a 6 degree angle of attack, Figure 3. Next to the wall the horseshoe vortex is again clearly visible as the lighter area on the wall next to the wing. The flow next to the wall separates from the wing to form a fishtail section. The fishtail is much larger for this case and large separated flow patterns are visible on the trailing edge of the wing. A large recirculation region, which merges with the separation area across the wing, is visible at the trailing edge corner.

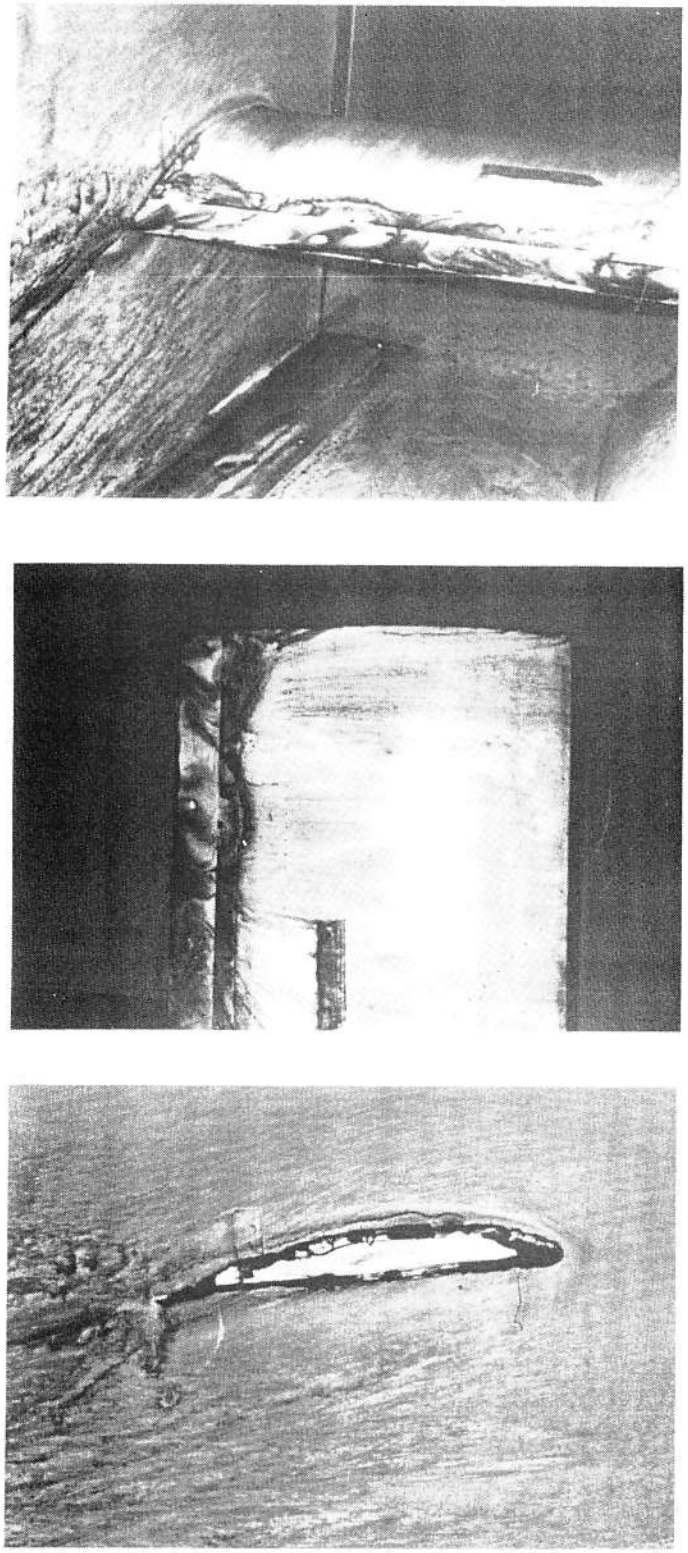

Figure 6: Wing-wall flow visualization.
$\alpha = 6^{\circ}$ Re = 500 000

The Navier-Stokes results are shown in Figures 7 and 8. It follows from the figures that the calculated resultts are qualitatively correct. The recirculation zones at the trailing edge of the wing-wall are correctly predicted. The laminar-turbulent position is not visible from the results and the position of flow separation is not predicted with high acuracy. The experimental results show the separation away from the wall to start at the trailing edge for the $\alpha = 2^{\circ}$ case and at 80 percent chord for the $\alpha = 6^{\circ}$ case. The simulation does not predict any separation away from the wall for the two cases. At the wall the separation is predicted to start at 60 percent chord for $\alpha = 2$ degrees while the experimental result shows this to be at the leading edge for both cases. The wall separation position for the $\alpha = 6$ degrees case is predicted at 50 percent chord.

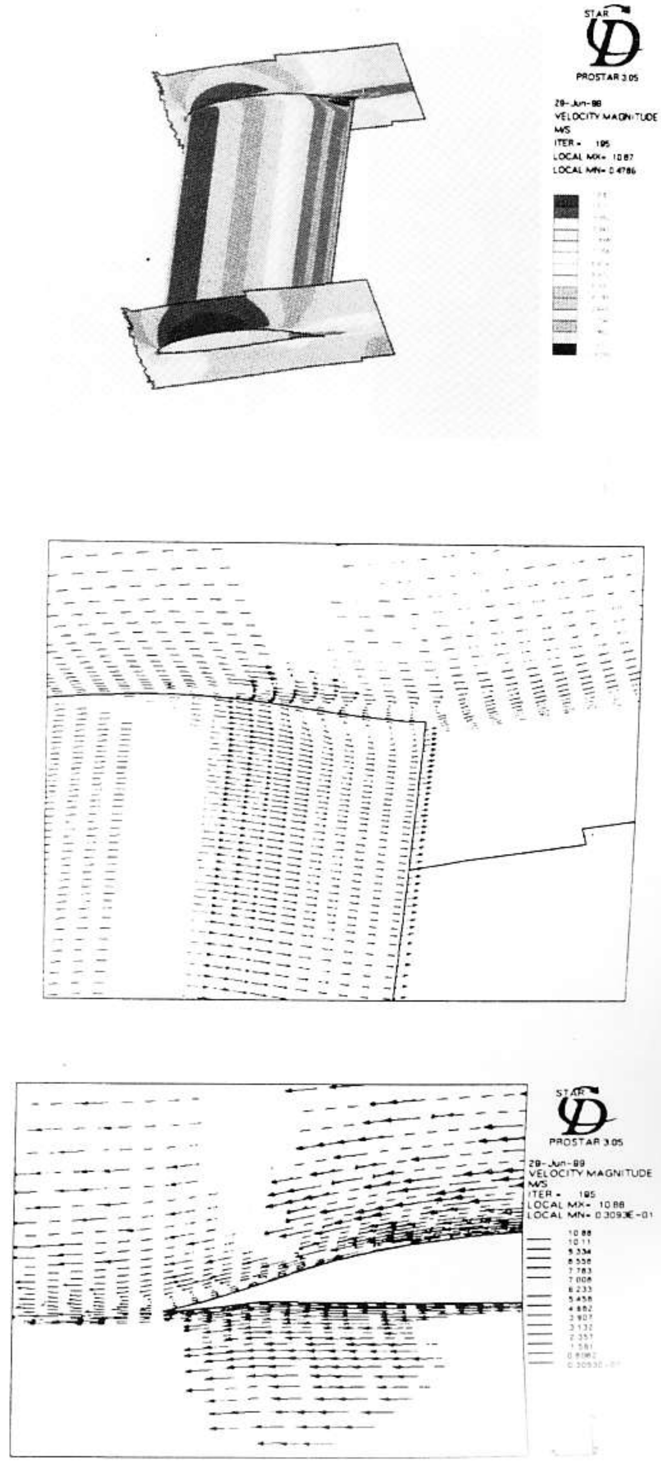

Figure 7: Navier-Stokes result for $\alpha = 2^{\circ}$, Re = 500 000

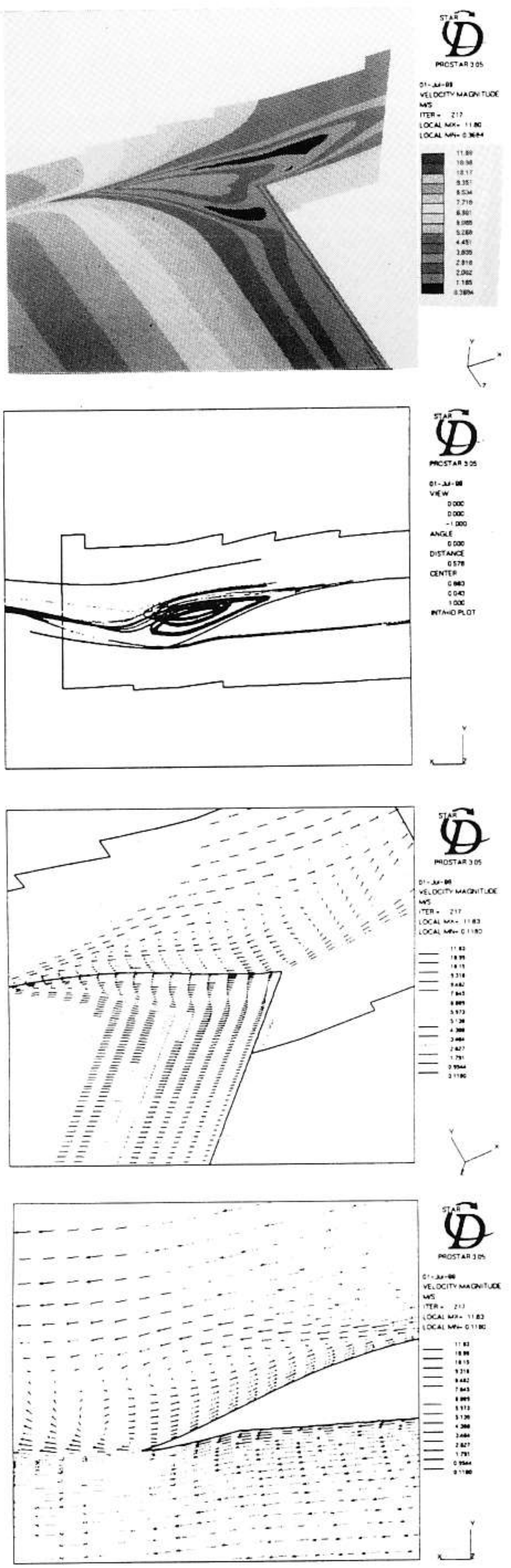

Figure 8: Navier-Stokes result for $\alpha = 6^{\circ}$ **Re = 500 000.**

The position of the fishtail wake is predicted accurately. Particle tracking is used in Figure 8 to show the position of the wing-wall vortex. This is in the expected position.

The pressure distributions on the wing at near-wall and far-wall positions are compared with the XFOIL results in Figure 9 and Figure 10.

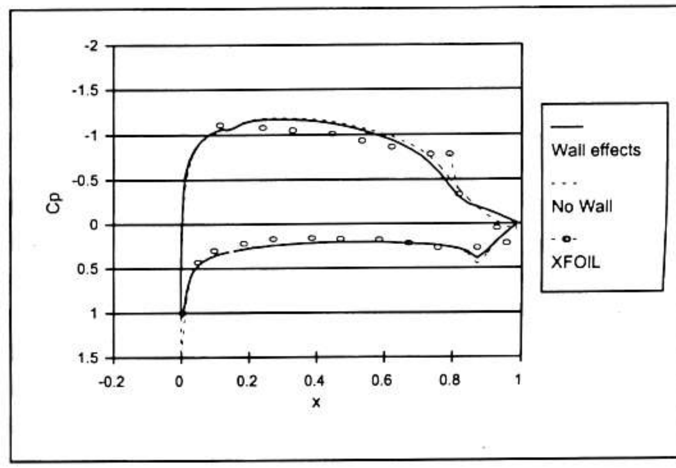

Figure 9: Pressure distribution for **a = 2°**

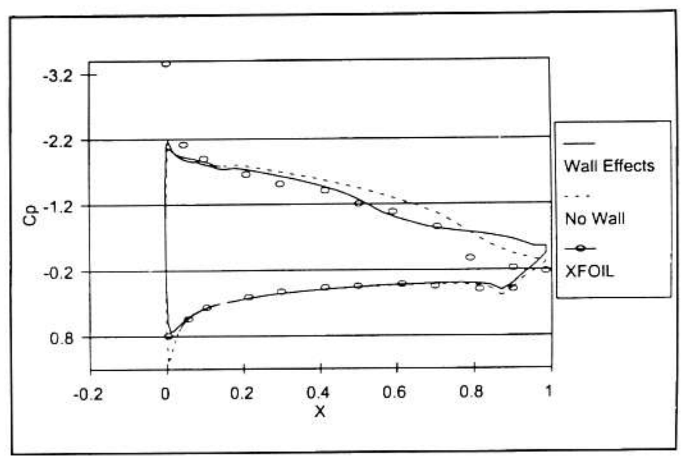

Figure 10: Pressure distribution for **a = 6°**

It follows from the figures that there are some differences compared to the XFOIL result, which was previously shown to be very accurate[7]. The Navier-Stokes solution does not show the laminar-turbulent transition. It is, however, clearly visible from the XFOIL result. This was expected[8] as it is known that these codes cannot accurately predict transition. The difference on the lower surface is small while the top surface shows the largest errors. The Navier-Stokes result fails to predict the suction peak on the airfoil leading edge for the $\alpha = 6^{\circ}$ case. The difference between the near-wall and far-wall pressure distribution is clearly visible in Figure 9.

The inaccuracies in the simulated results can be attributed to the inaccuracy of the turbulence model. The turbulence model used, is a very simple k-∈ model. More complex version of this mdoel which is more suited for the solution of exterior flow simulations as this, does exist but was not evaluated at this stage. The developers of the codes are working on a Reynolds stress model which is generally believed to give more accurate answers.

CASE 2: THE FLOW AROUND A WING-BODY GEOMETRY

The second case modeled is the wing-body junction shown in Figure 11. The fuselage is constructed from the Althaus shape two data[9] which is increased in length for pilot safety. The maximum fuselage height is 0.7m with a 0.6m width and a length of 6.5m. The wing section is the AS97-144/14, Figure 2, at the tip and a modified version at the root more suitable for the turbulent flow conditions present at the wing-fuselage junction.

THE COMPUTATIONAL GRID

A structured computational grid of approximaely 210,000 cells was used to model the problem. The grid is again a rectangular block with the wing-body shape cut out from it. It is refined near the wing and at the junction to ensure high accuracy in the areas where separation is expected.

The grid was developed from a purposely developed CAD interface for glider geometries on the drawing package CadKey. The interface allows very quick and easy development of the geometry and produces the spline and vortex data required to develop the grid. The grid was developed as a user program for Flo++ and the development took approximately two days. With this program it is now possible to reproduce new grids from the CAD data in a few minutes allowing very quick changes to the geometry.

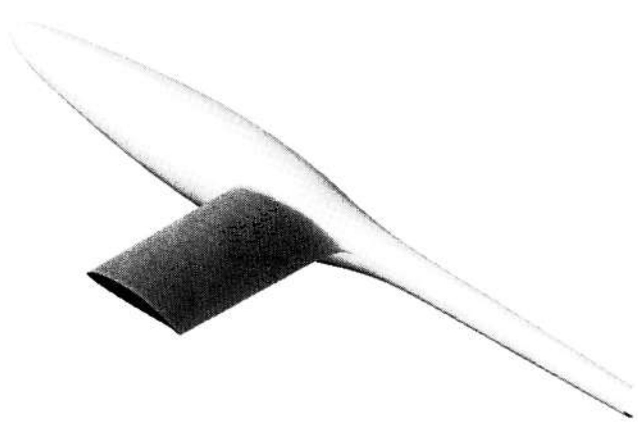

Figure 11: Wing-Fuselage geometry

As can be seen from Figure 12, only half the fuselage was modeled around the lateral symmetry plane. This allows for a smaller computational grid with the consequent computational time savings. Pressure boundaries were used at the top and bottom of the grid with a symmetry boundaries at the sides. A constant velocity of 25m/s was used across the inlet.

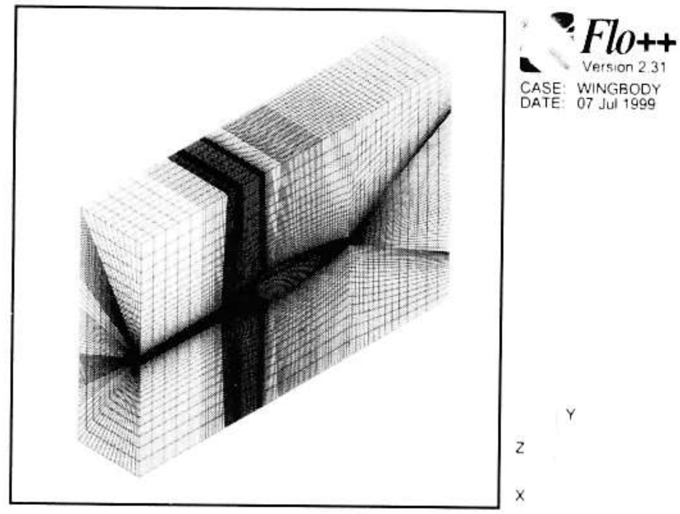

Figure 12: The computational grid

RESULTS

The calculation was done on a 300Mhz Pentium was 256 MB RAM and it took 4 h to converge.

Figures 13 and 14 shows the surface pressure distribution over the junction. The pressure distribution away and at the junction does not differ much. The Navier-Stokes method does not predict any separation on the wing away and at the junction. This might be the result of the effects being very small and the grid not refined enough to capture the results. A high pressure spot is also shown on the fuselage behind the wing in line with the wake of the wing.

Figure 15 shows the velocity vector plot just behind the trailing edge next to the fuselage. A small vortex is visible next to the fuselage just above the trailing edge. This is the expected position of the vortex. The grid, however, needs to be refined further to completely capture the vortex.

The results of this investigation show that the Navier-Stokes method can be used to qualitatively investigate the flow in complex regions. It is however very dependent on a very fine computational grid to capture the required effect, which in turn lengthens then calculations times, sometimes to a point where the method is useless as a practical design code.

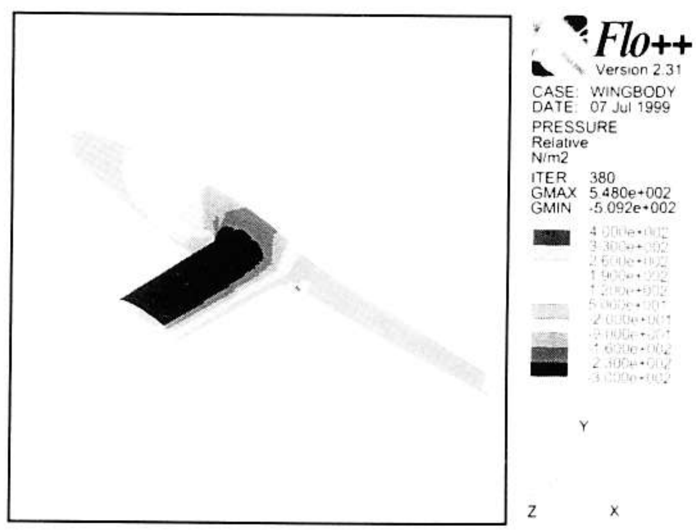

Figure 13: Surface pressure distribution

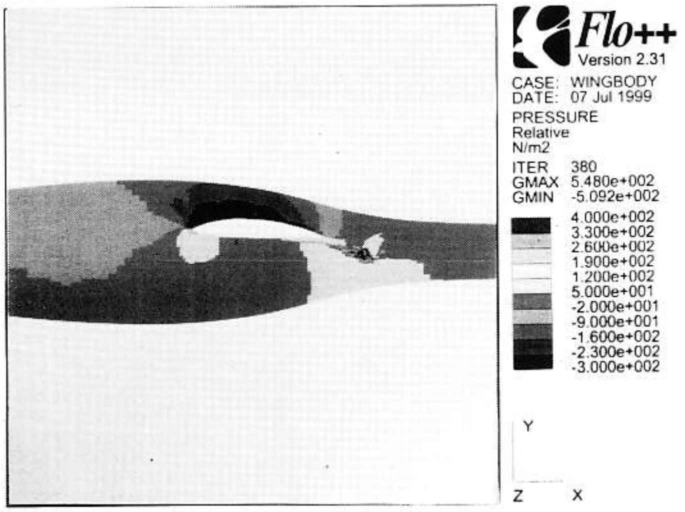

Figure 14: Surface pressure distribution

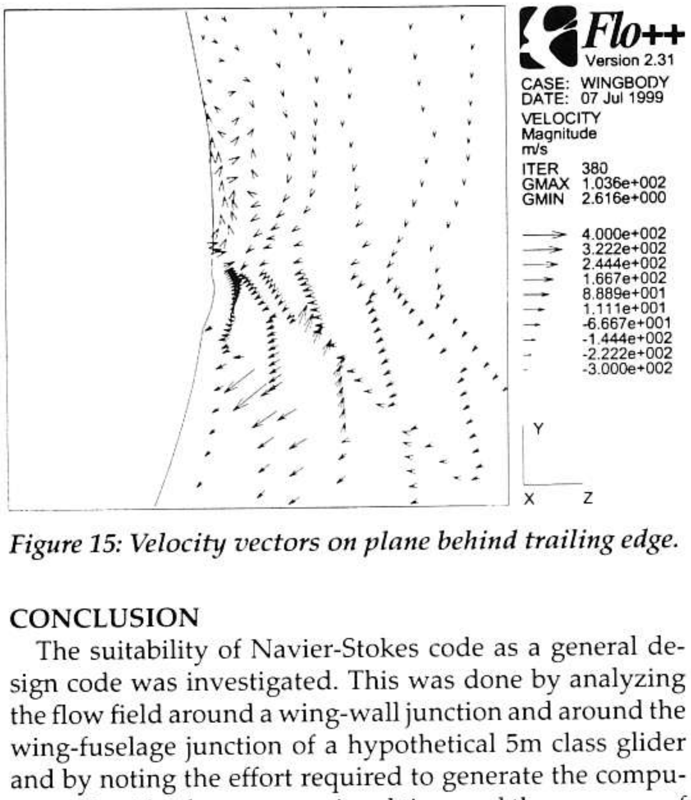

Figure 15: Velocity vectors on plane behind trailing edge.

CONCLUSION

The suitability of Navier-Stokes code as a general design code was investigated. This was done by analyzing the flow field around a wing-wall junction and around the wing-fuselage junction of a hypothetical 5m class glider and by noting the effort required to generate the computational grids, the computational time and the accuracy of the results. It is found that the problem of grid generation can efficiently be solved by using the purposely developed CAD interface which allows glider geometries to be generated easily. The structured computational grid can then be devleoped from the CAD model without undue effort. The solution time depends on the computer capacity, memory available as well as the grid size. A typical wing-fuselage junction consists of 320,000 cells which took 4.5 h to solve on a 300 Mhz PC with 256 MB Ram while it took only about 2.5 h oa Dec Alpha workstation.

The solution was found to show the general flow characteristics expected around a wing-wall/wing-body geometries. The prediction of the laminar-turbulent and separation points are generally too in accurate for the design of high performance sailplane configurations. This is the result of the fact that the k-∈ turbulence model is not very suitable for complex flow field as was found here. The developers of the code are working on the introduction of newer turbulence models such as the Reynolds stress model, which might improve the accuracy. The problem might be overcome with a user coded empirical transition model which will force transition on the correct positions.

The time problems associated with Navier-Stokes codes are being eased with the availability of more powerful computers. This has reached the stage where a sailplane designer might consider the use of a commercially available Navier-Stokes code as an alternative to the current crop of panel codes. Once a robust turbulence model has been developed, these methods will be usable as a practical design code.

REFERENCES

1. Boermans, L.M.M., Nicolosi, F., Kubrynski, K., Aerodynamic design of high-performance sailplane wing-fuselage combinations. ICAS 98-2.9.3, 1998.

2. Davenport, W.J., Simpson, R.L., Flow past a wing-body junction - Experimental evaluation of turbulence models. AIAA Journal, 30 No. 4, 1992, p. 873-881.

3. Jameson, A., Full potential, Euler and Navier-Stokes Schemes. Applied computational aerodynamics. Ed. P., A., Henne, 1990, p. 39-75.

4. Hallback, M., Johansson, A.V., Burden, A.D., The basics of turbulence modeling. Turbulence and transition modeling. Ed. Hallback et al. 1996, p. 84.

5. Boermans, L.M.M., van Garrel, A., Design and wind tunnel test results of a flapped laminar flow airfoil for high-performance sailplane applications. ICAS-94-5.4.3.

6. Drela, M., Elements of airfoil design methodology. Applied computational aerodynamics. Ed. P/., A., Henne, 1990, p. 167-189.

7. Roberts, P., Die konstruksie van 'n windtonnel model en die ondersoek na die lugdinamiese eienskappe van die AS-97-129/14 vlerkprofiel. Final year thesis, Potchefstroom University for CHE, 1997.

8. Savill, A.M., One Point closures applied to transition. Turbulence and transition modeling. Ed. Hallback et al. 1996, p. 262.

9. Althaus, D., Wint tunnel measurements on bodies and wing-body combinations. Motorless flight research. NASA CR-2315, 1972.